SMT Troubleshooting

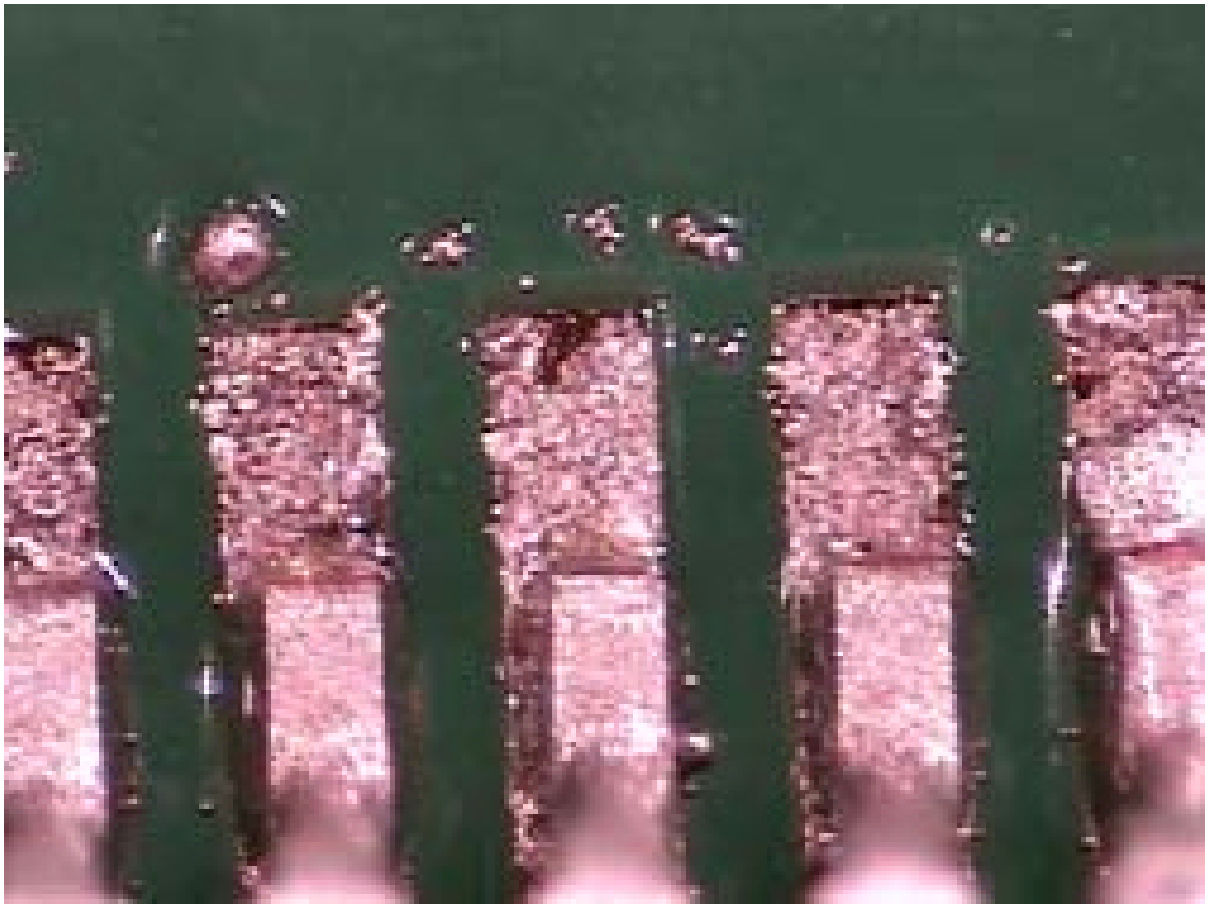
Typical SMT Problems

For additional process solutions, please refer to the AIM website troubleshooting guide

- Solder Balling
- Solder Beading
- Bridging
- Opens
- Voiding
- Tombstoning
- Unmelted Paste
- Disturbed Joints
- Excessive Fillet
- Non Wetting
- Dewetting

Solder Balling

- Recognized by:
  - Numerous tiny solder balls trapped along the peripheral edge of the flux residue after reflow
  - Balls stuck around fine pitch lands and solder mask



Solder Balling

- Possible Cause:
  - Oxidized paste- Will be identified by halos gray in appearance around the solder balls or joints.
    - Was paste shipped refrigerated next day air? Did it spend an extended time in a hot receiving area? Was old paste returned to a new jar? Was the paste put back in a refrigerator after it had sat out, causing condensation?
- Remedy:
  - Run fresh paste from a different lot under same conditions and see if solder balls go away.

Solder Balling

- Possible Cause:
  - Paste has been on the PCBs too long before reflow.
- Remedy:
  - Try running a PCB with fresh paste and see if the problem disappears

Solder Balling

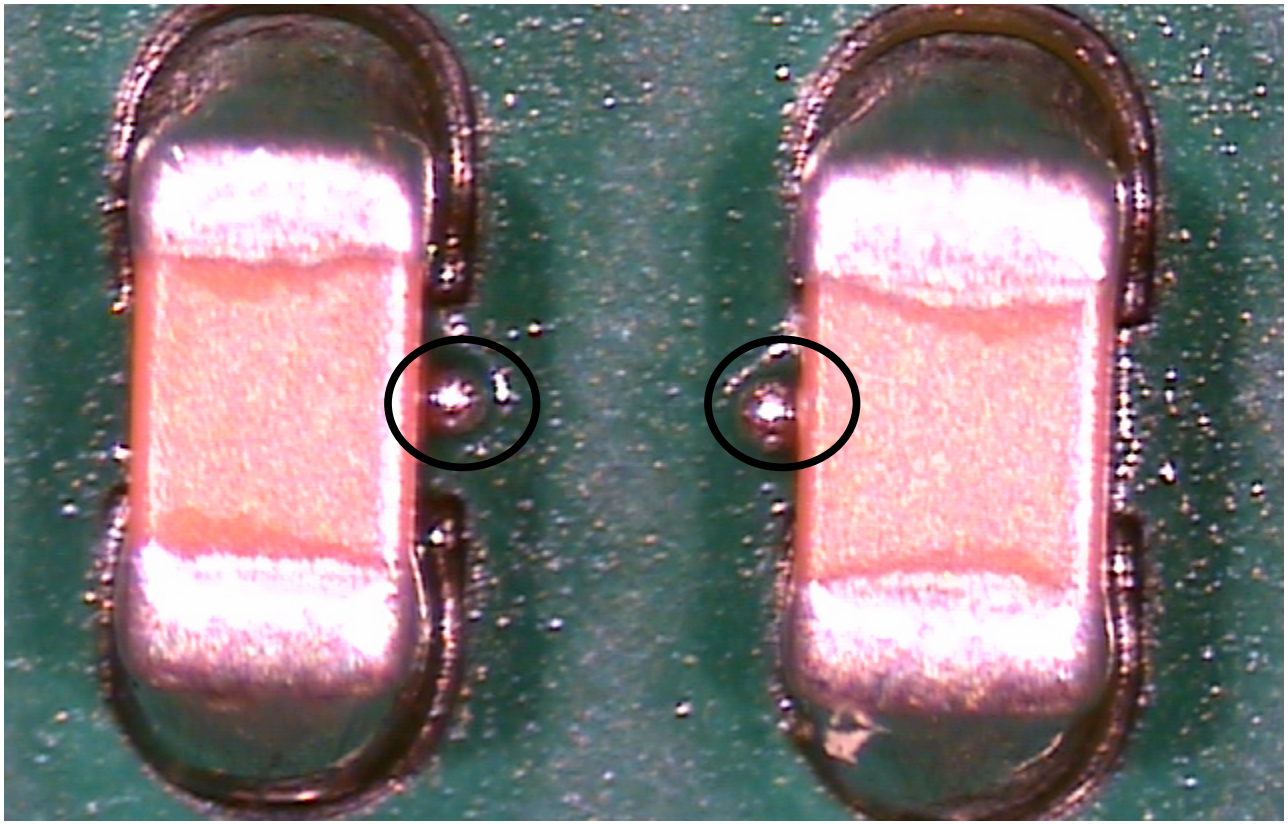
- Possible Cause:
  - Reflow profile too slow on ramp up
- Remedy:
  - Check the ramp rate in the first 90 seconds of the profile.
  - Run recommended profile and see if the problem persists

Solder Spattering

- Possible Cause:
  - Reflow profile too rapid on ramp up. Too rapid ramp up to reflow temperature will not allow the volatiles in the paste to be driven off before paste becomes molten. A combination of volatiles and molten solder will result in solder spatter.
- Remedy:
  - Run a slower ramp up profile to give the volatiles time to vaporize

Solder Beading

- Recognized by:
  - Single large solder balls that are next to a component or component pad
  - Typically found by discrete components.



Solder Beading

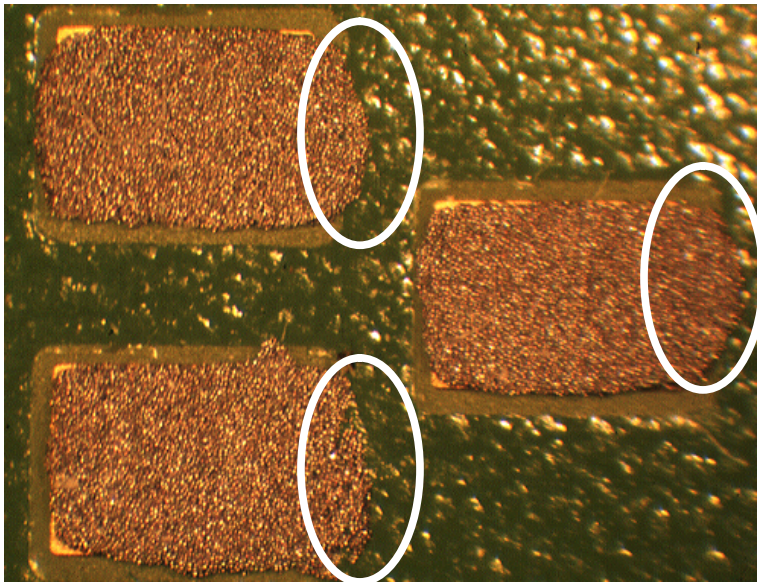
- Possible Cause:
  - Reflow profile ramp up too slow causing capillary action to draw the unreflowed paste away from the pad on which it was deposited to a place under the component. It reflows there forming a bead of solder that comes out to the side of the component
- Remedy:
  - Use a rapid ramp up of 1.5 to 2.5 degrees C/sec.

Solder Beading

- Possible Cause:
  - Too low squeegee pressure leaving too much paste behind.
- Remedy:
  - Increase squeegee pressure

Solder Beading

- Possible Cause:
  - Excessive amount of solder paste printed on component pads. At reflow excess molten solder is squeezed out the side of the component



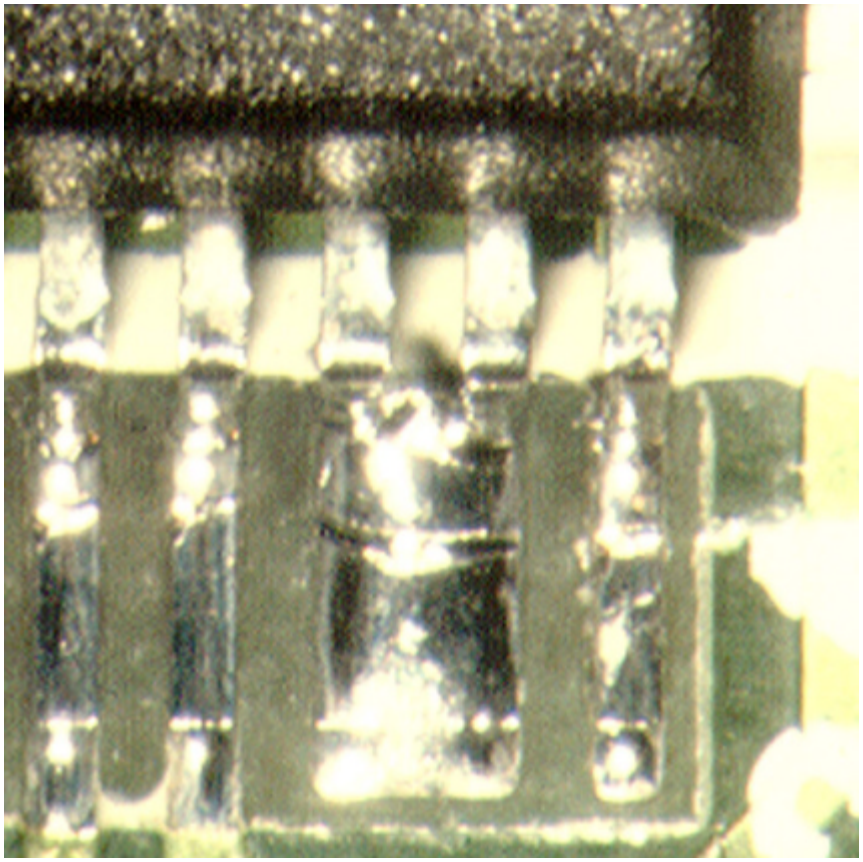
- Remedy:
  - Reduce the amount of solder paste printed on the pad.

Solder Beading

- Possible Cause:
  - Paste bleeding under the stencil. This paste is transmitted to the PCB outside the aperture dimensions. When reflow occurs it may remain on the PCB adjacent to the component aperture in solder bead form.
- Remedy:
  - Check for excess squeegee pressure or improper gasketing

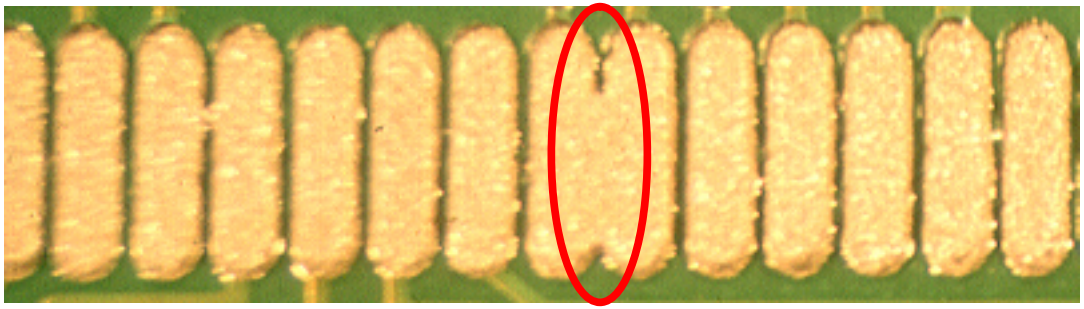
Bridging

- Recognized by solder running from one component contact to another. This could result in a short circuit.



Bridging

- Possible Cause:
  - Cold Slump
- Remedy:



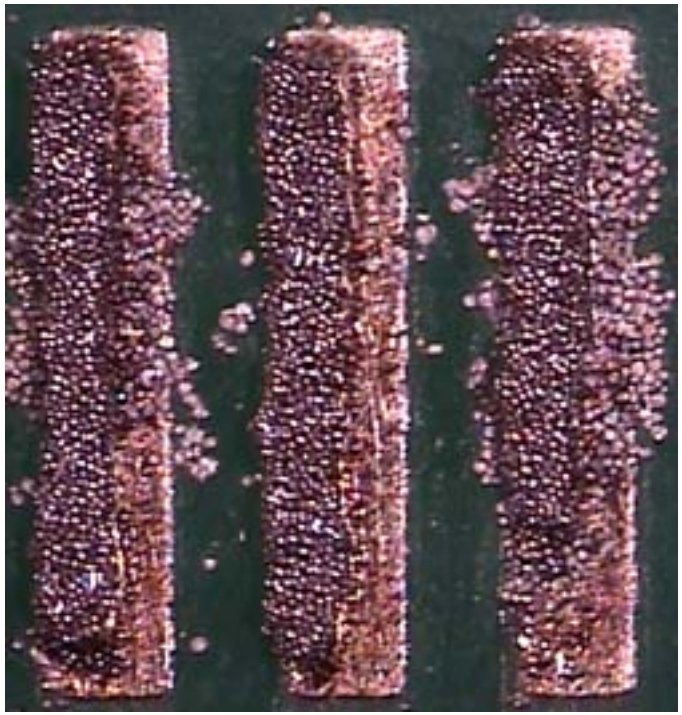
  - Cold slump is generally due to paste being the incorrect viscosity or due to a too rapid squeegee speed over shearing the paste and degrading its thickeners.
  - Check the viscosity of the paste. If correct, lower print speed and run fresh paste to see if problem persists.
  - Might also be humidity related. Try running fresh paste.

Bridging

- Possible Cause:
  - Hot Slump
- Remedy:
  - Hot Slump occurs when the paste deposits settle or spread out during the ramp up cycle of the reflow profile and spread over to an adjacent pad.
  - Check the overall profile length.
  - Check the ramp rate in the first 90 seconds of the profile.
  - Shorten the duration of ramp up and see if the bridging disappears.
  - Change the profile type.

Bridging

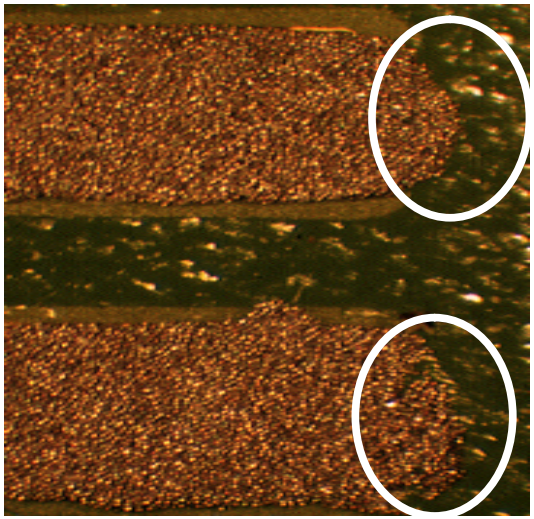
- Possible Cause:
  - Excess squeegee pressure causing paste bleed out.



- Remedy:
  - Reduce squeegee pressure. Insure on contact printing and proper gasketing.

Bridging

- Possible Cause:
  - Evidence of excessive solder paste being deposited on the pads between which the bridge is occurring. Upon reflow, the excess solder may flow over to an adjacent pad, causing a bridge.



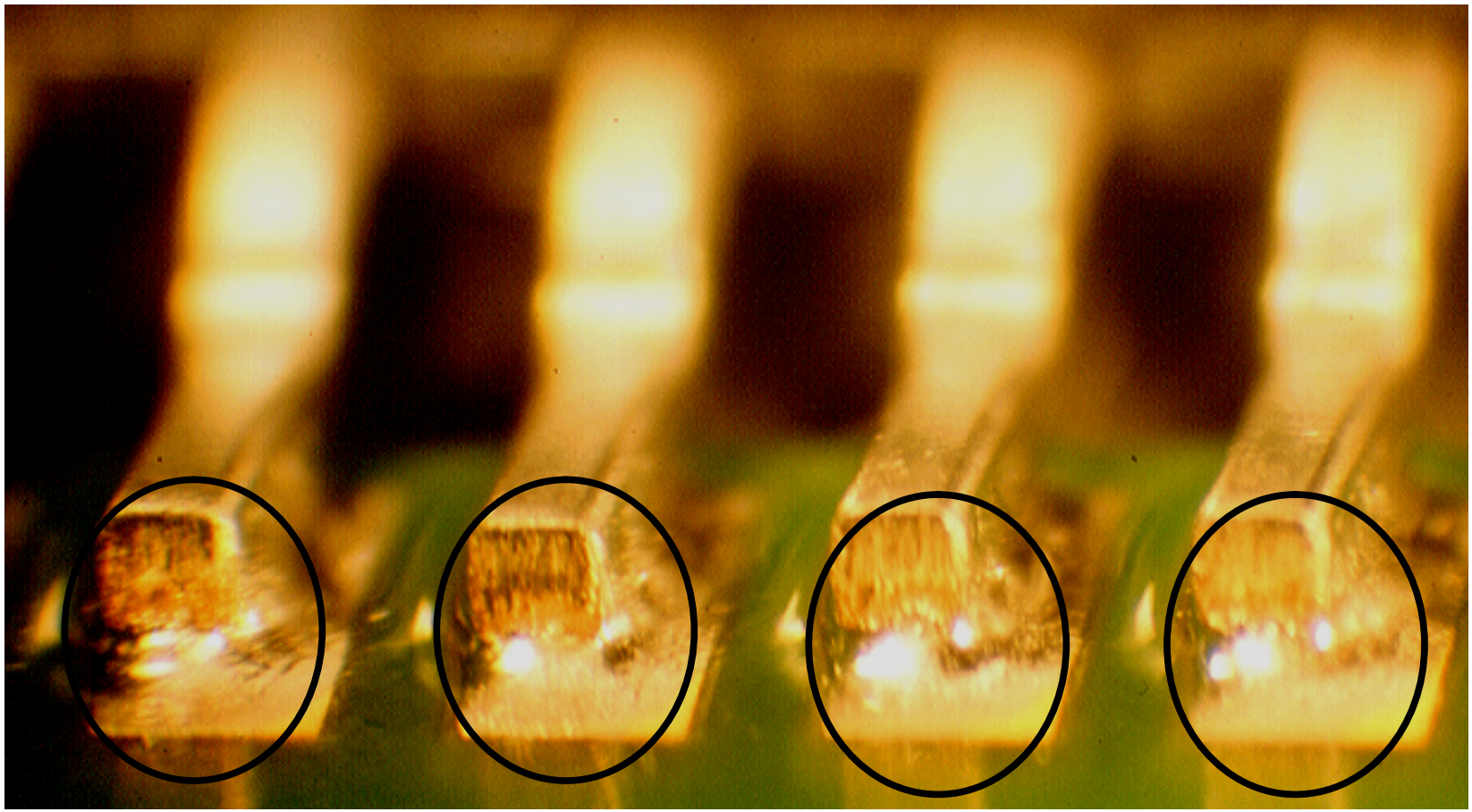
- Remedy:
  - Reducing the aperture dimensions or reducing stencil thickness will eliminate excessive solder.

Bridging

- Possible Cause:
  - Excessive part placement pressure, insufficient board support (in printer and/or placement) or incorrect alignment.
- Remedy:
  - Check and correct placement settings as necessary.

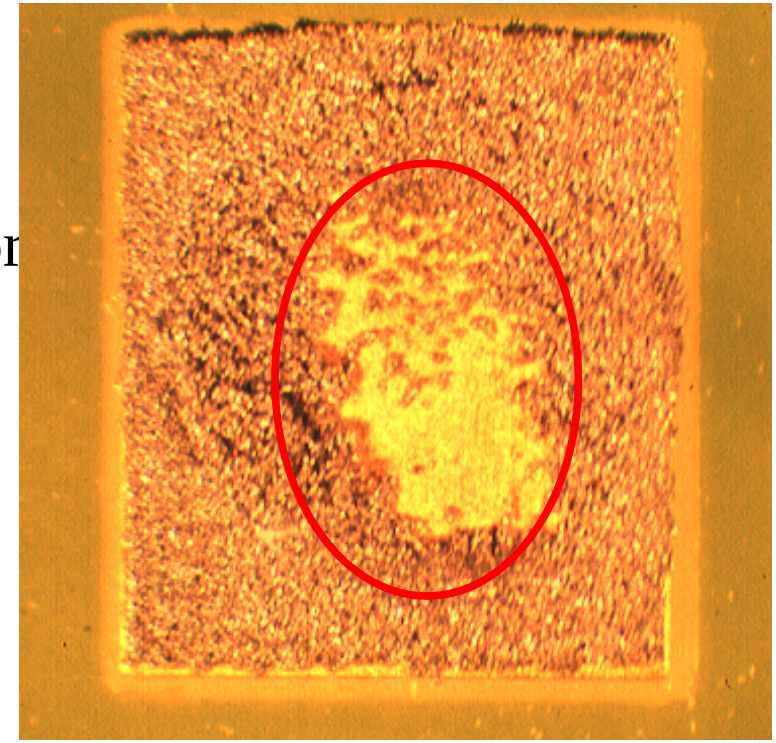
Opens or Insufficient Fillet

- Recognized by insufficient solder to make a complete bond between the lead and the pad



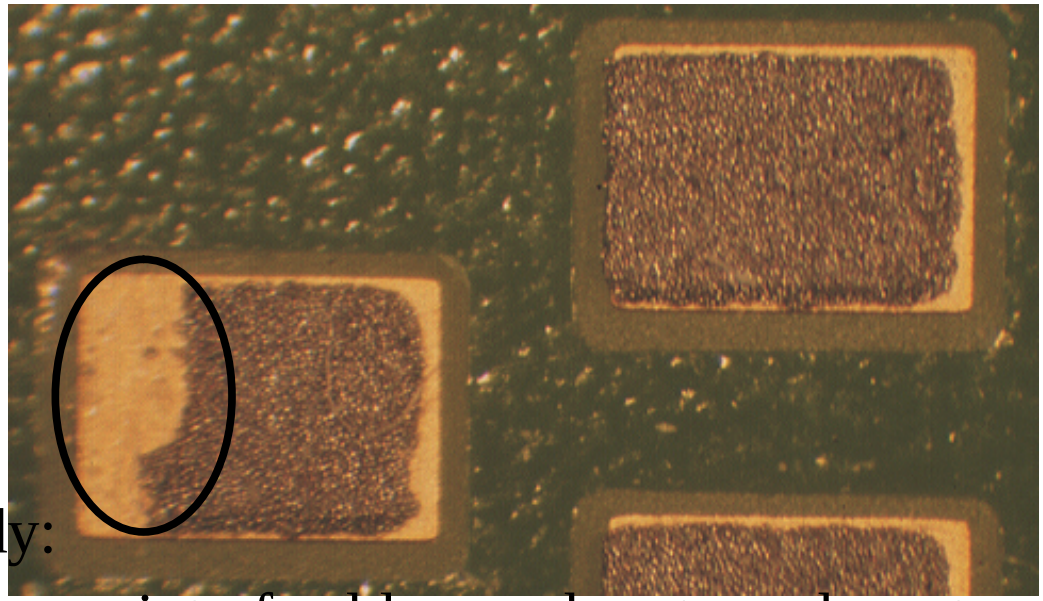
Opens or Insufficient Fillet

- Possible Cause:
  - Scooping due to excessive squeegee pressure. The excess pressure forces the edge of the squeegee to go into the aperture, scooping out some of the paste intended to be deposited.
- Remedy:
  - Decrease pressure or use a harder durometer polypropylene or metal squeegee.



Opens or Insufficient Fillet

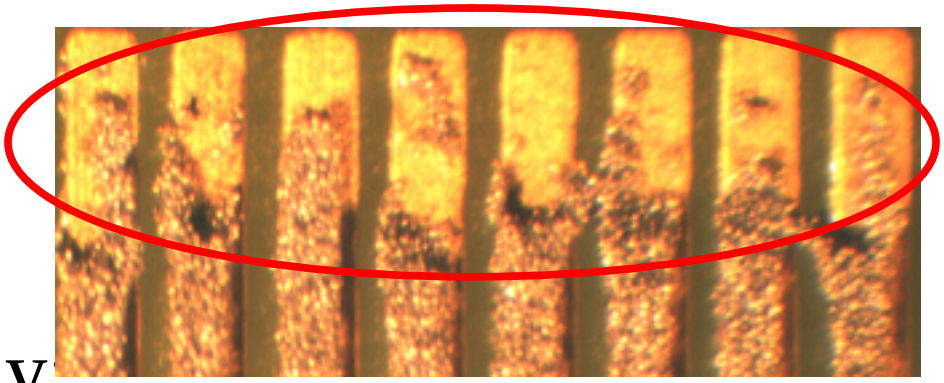
- Possible Cause:
  - A foreign body or dryed up paste in the aperture, possible over print of solder mask onto the effected pad,



- Remedy:
  - If overprint of solder mask onto pad, customers only recourse is to resolve the situation with their PCB supplier. Regular examination of and cleaning of stencils should eliminate the other problems.

Opens or Insufficient Fillet

- Possible Cause:
  - Squeegee speed to fast to fill the apertures.



- Remedy:
  - Slow down squeegee speed and see if this corrects the problem

Opens or Insufficient Fillet

- Possible Cause:
  - Solder paste of too too high a viscosity, not allowing it to roll on the stencil and/or fill the apertures
- Remedy:
  - Check viscosity and metal content to see if these meets the manufacturer’s specifications.

Opens or Insufficient Fillet

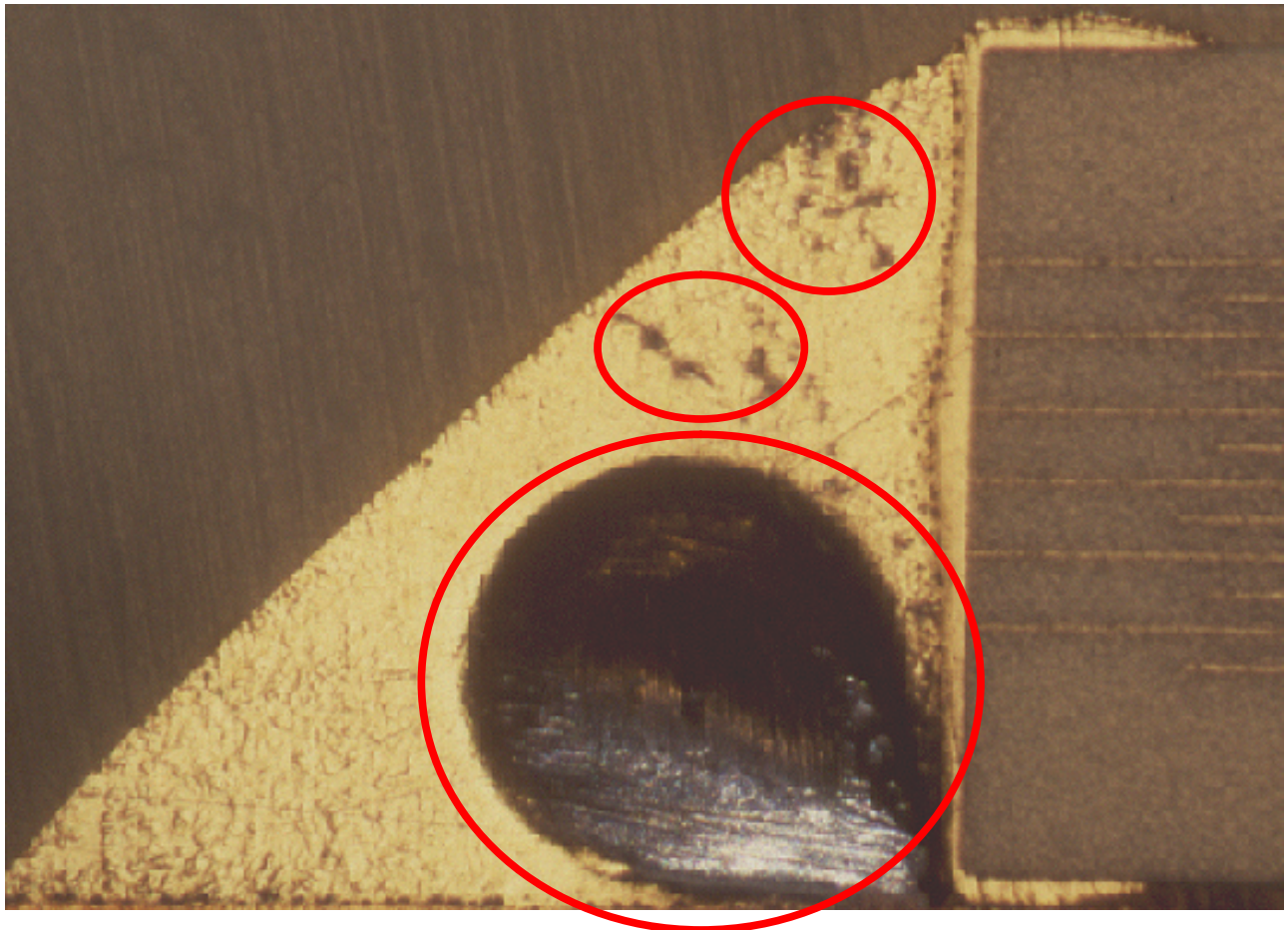
- Possible Cause:
  - Solder paste wicking up leads as a result of uneven or excess heating.
- Remedy:
  - Reduce the reflow peak temperature.
  - Increase bottom-side heat.

Opens or Insufficient Fillet

- Other possible causes:
  - Separation speed to fast
  - Separation distance to small
  - Poor stencil cut
- Inspect these settings one by one

Voiding

- Recognized by the appearance of many tiny or a few large “bubbles” in the joint. These may be air or flux entrapment.
- Commonly found through X-ray or cross inspection.

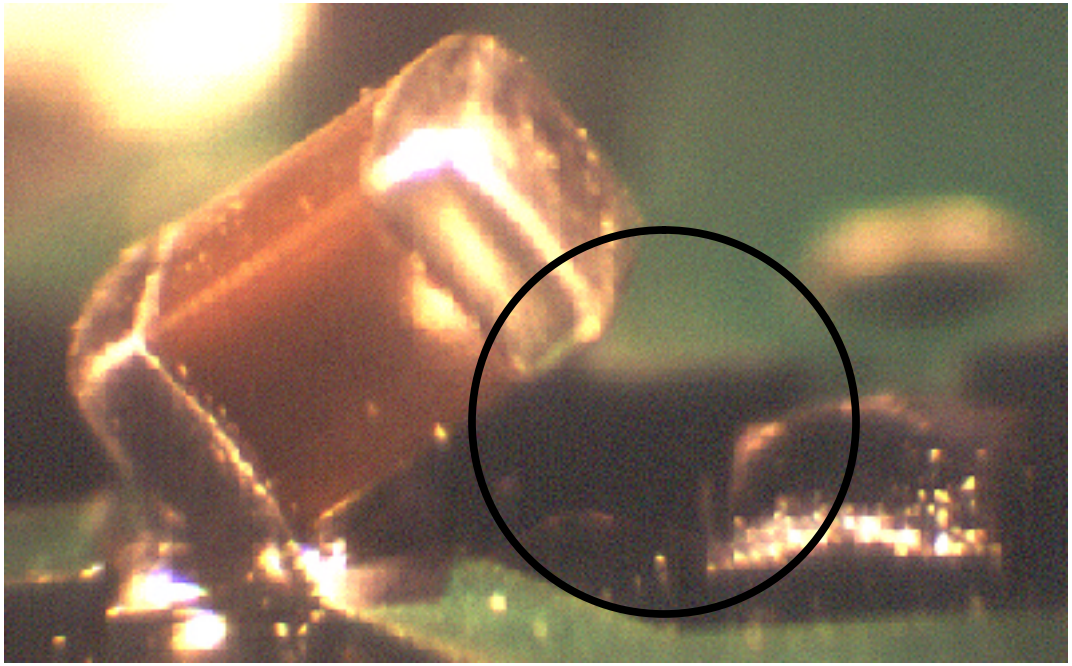


Voiding

- Possible Cause:
  - Too low peak temperature.
  - Too short time at temperature.
  - Soak too long.
  - Soak too hot.
  - Excessive ramp rate.
- Remedy:
  - Inspect the reflow profile and adjust accordingly.

Tombstoning

- Recognized by chip type components standing on end after reflow. The root cause of this is unequal soldering forces being exerted on each end of the component. This is very common in vapor phase reflow.



Tombstoning

- Possible Cause:
  - Unequal heat sink, such as a ground plane that draws heat from one end of the component.
- Remedy:
  - Increase duration of soak cycle to insure all parts of the PCB are at uniform temperature.

Tombstoning

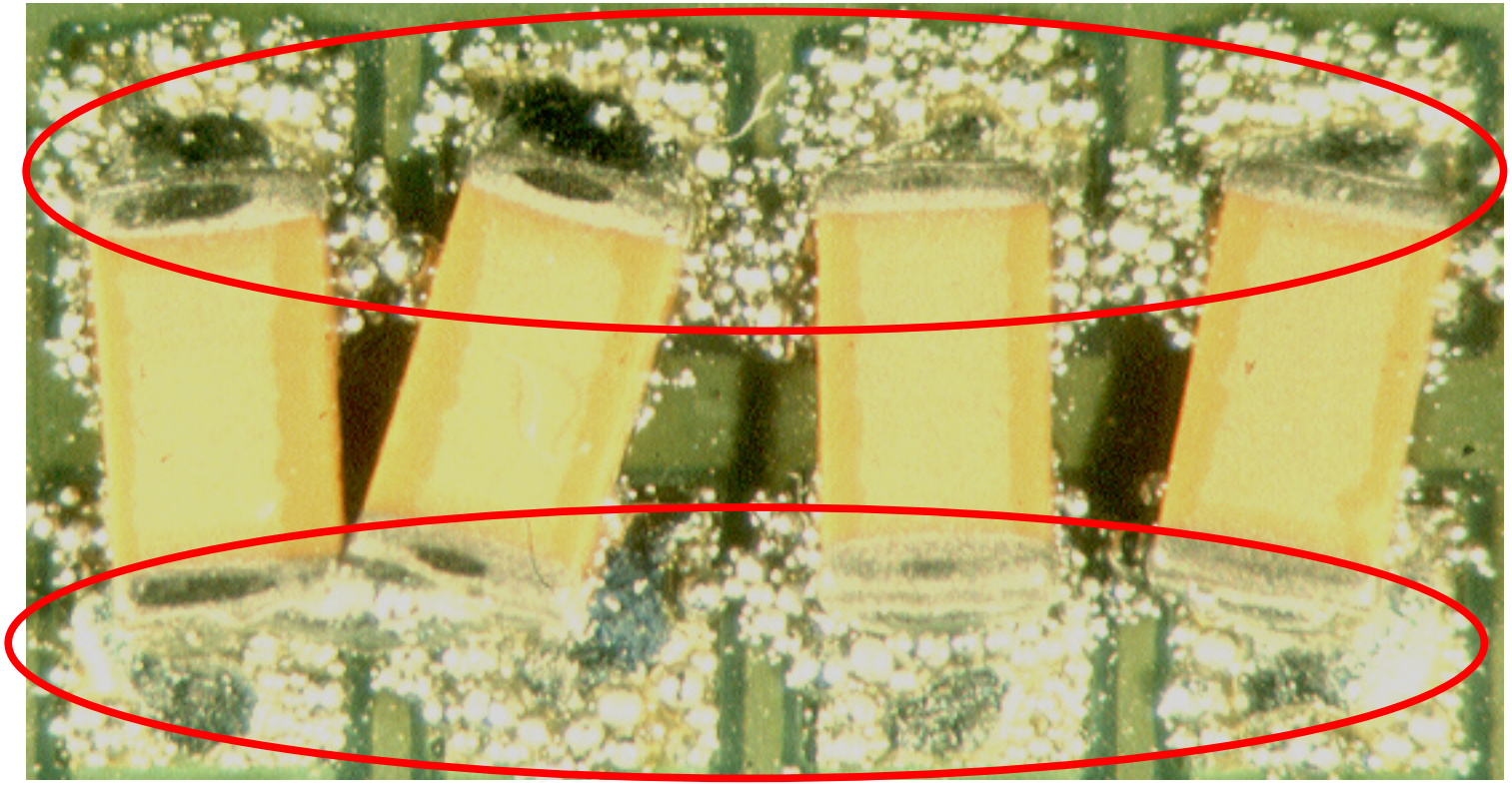
- Possible Cause:
  - Solderability of the components may be marginal.
- Remedy:
  - Ask if customer tests incoming components and PCB pads for solderability.
  - If component solderability is marginal, the slightest inequality in placement on the pad or heat delivered to each pad may result in tombstoning. Components should be tested for solderability.

Tombstoning

- Possible Cause:
  - Unequal placement on the pads prior to reflow. This will result in an imbalance of soldering forces, often resulting in tombstoning.
- Remedy:
  - Alter placement position to insure equal placement on pads.

Unmelted Paste

- Recognized by unreflowed paste on the component pads.



Unmelted Paste

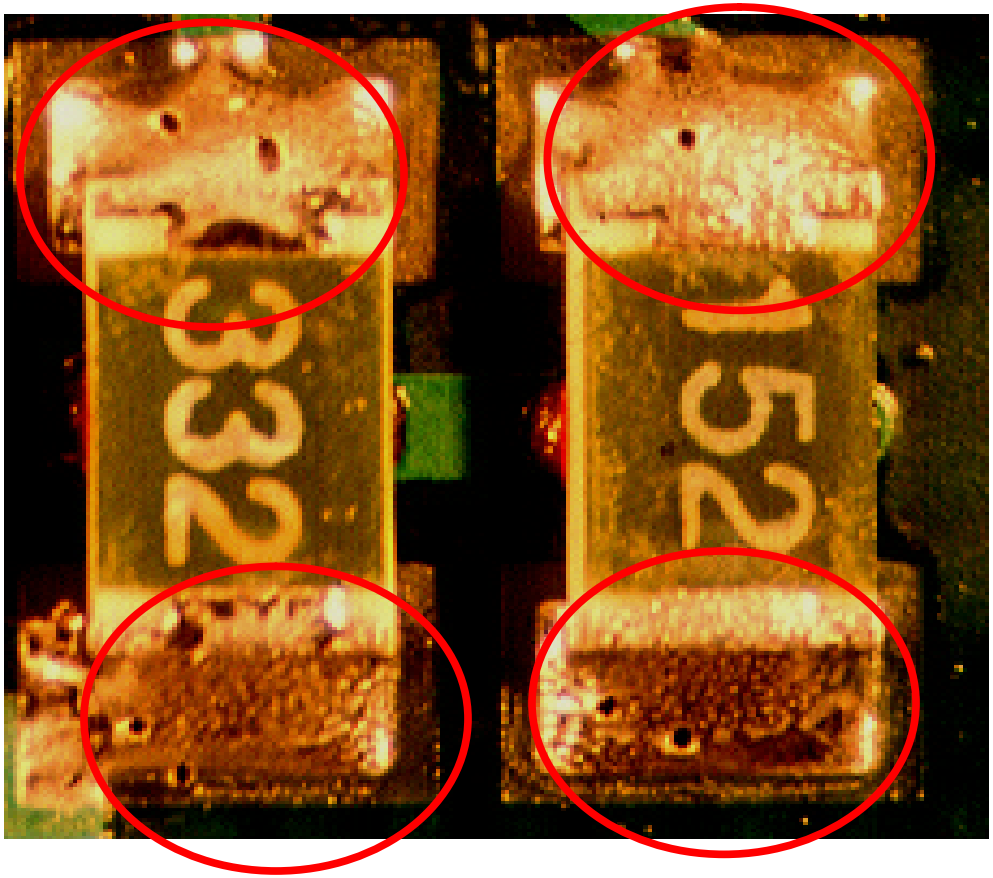
- Possible Cause:
  - Too low of a reflow profile peak.
  - Too short of time at temperature.
  - Too long a soak zone.
  - Too hot a soak zone.
- Remedy:
  - Correct the profile.

Unmelted Paste

- Possible Cause:
  - The wrong alloy/reflow profile is being used.
- Remedy:
  - Verify and adjust accordingly.

Disturbed or Cold Joint

- Recognized by dull, rough appearance of solder in an alloy that is normally bright and shiny. Testing of the joint will result in findings of reduced mechanical strength as a result of little or no intermetallic bond.

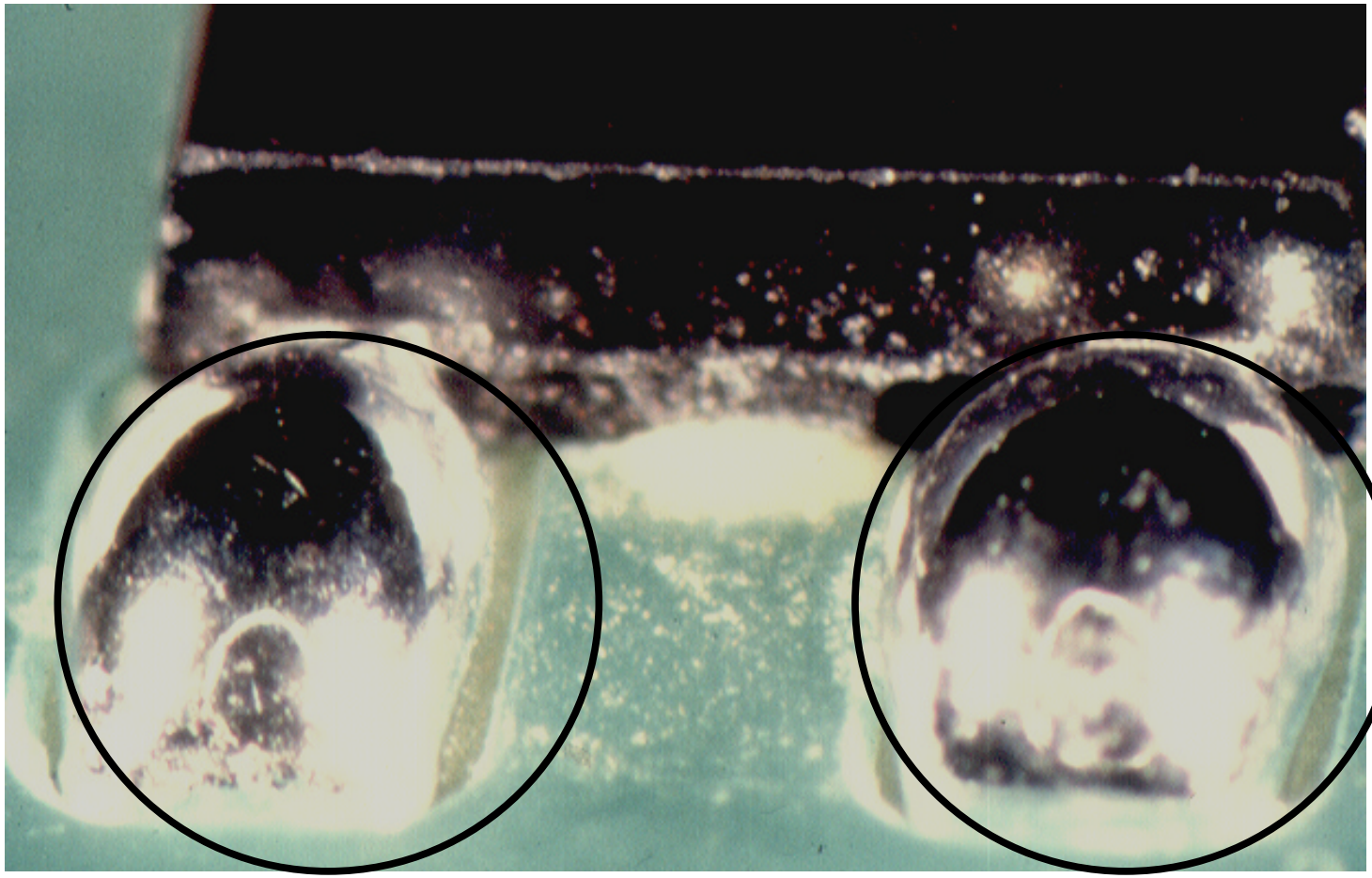


Disturbed or “Cold” Joint

- Possible Cause:
  - A source of vibration that could be transmitted to the PCB after reflow of the solder paste while the solder is still in a liquid state.
    - This often is a jerking belt or conveyor in the oven
  - Too long of a cooling cycle.
  - Alloys with high percentages of lead are naturally dull.
- Remedy:
  - Fix the source of vibration and/or adjust the reflow cool down profile so that the joints are solidified upon exit.

Excess Solder on Fillet

- Recognized by bulbous convex appearance of joint where the outlines of the leads are obscured by the quantity of solder on them.

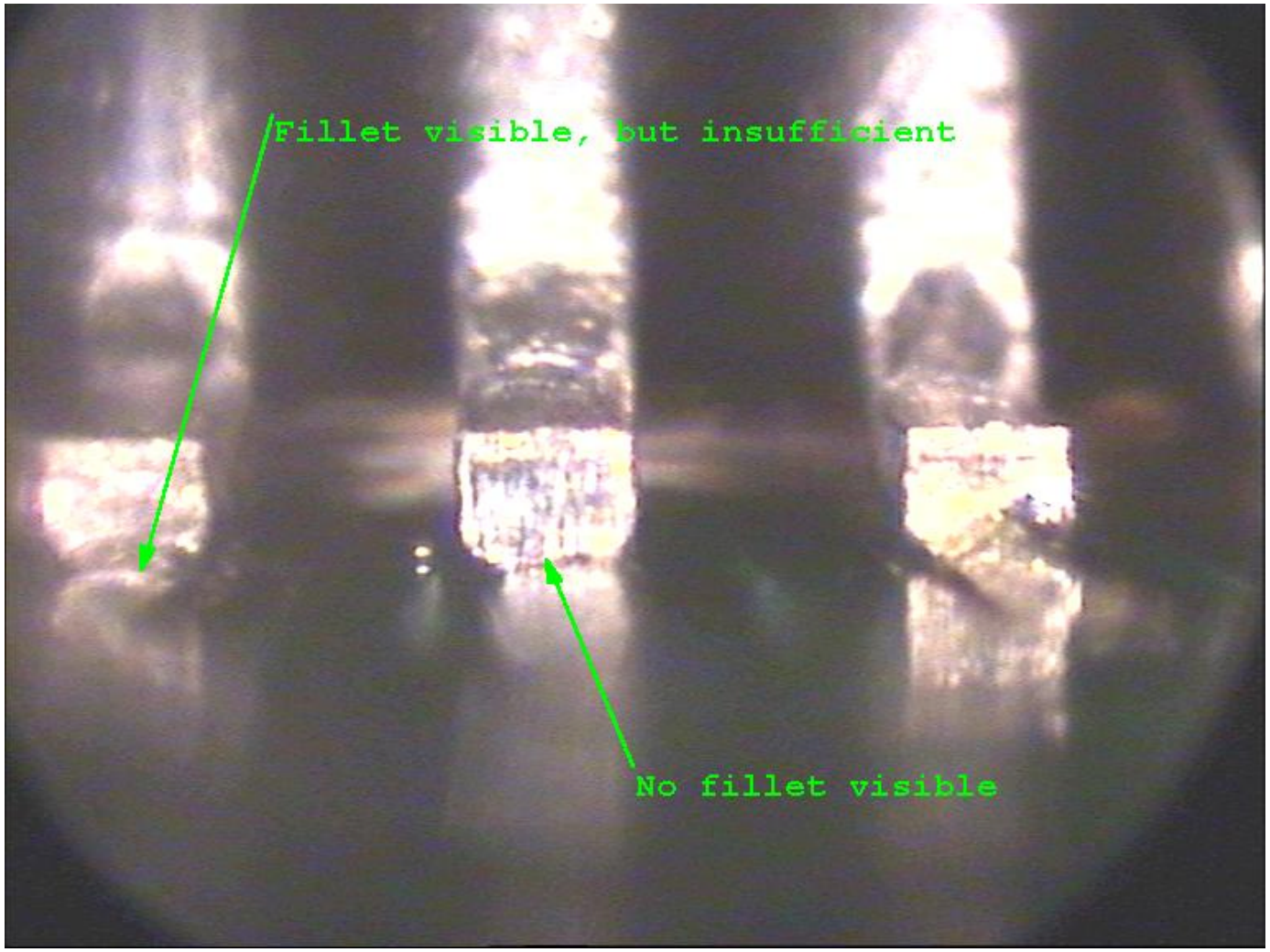


Excess Solder on Fillet

- Possible Cause:
  - Conditions which have caused too much paste to be deposited on the pad of the joints with excessive solder on them.
    - Stencil thickness
    - Squeegee Pressure
    - Snap-Off
    - Bleed Out
    - Aperture Size/Design
- Remedy:
  - Reduce the amount of paste printed on the affected areas.

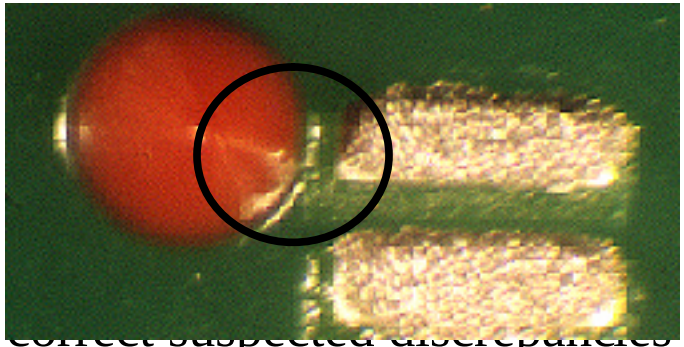
Non-Wetting

- Recognized by an insufficient solder fillet.



Non-Wetting

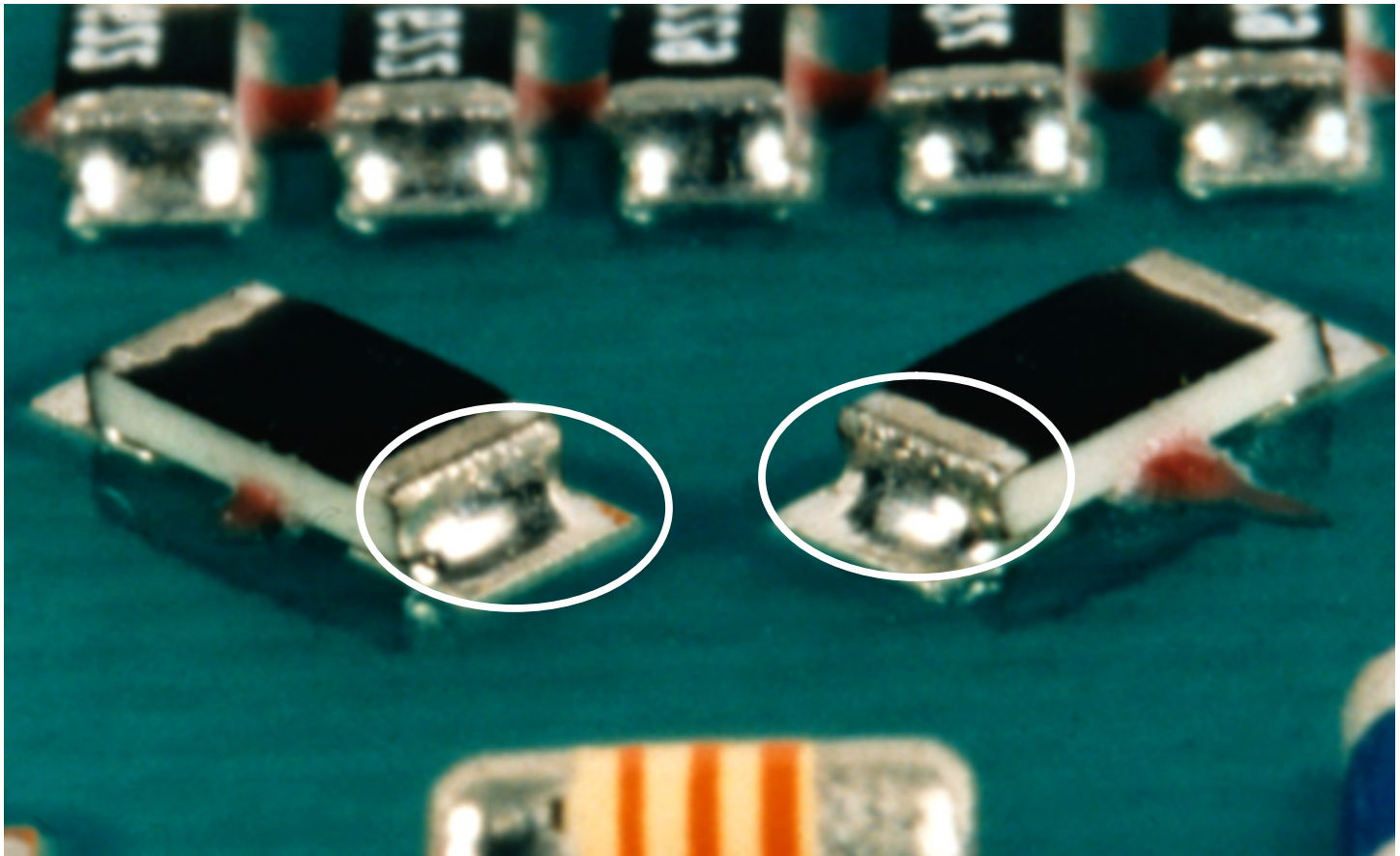
- Possible Causes:
  - Grease, oil or dirt on the surface to be soldered
  - Bleeding or misregistered soldered mask
  - Excessive, incorrectly placed, or excessive adhesive
  - Surfaces too heavily oxidized for solder paste being used
  - Old boards and/or components being used that have suffered from copper migration
  - Scratched pad resulting in exposed copper.
  - Poor activation of paste resulting from paste chemistry or reflow profile.



- Remedy:
  - Investigate each possible cause and correct suspected discrepancies one at a time until solderability is restored

Dewetting

- Recognized by metal wetting initially, then pulling back to form droplets of solder on the surface. Dewetting will give the surface a tinned appearance.



Dewetting

- Possible Causes:
  - Contamination of surface by abrasives
  - Poor plating
  - Poor hot air solder leveling during PCB manufacture